Coupled RANS-LES Computation of a Compressor and Combustor in a Gas Turbine Engine

J. U. Schlüter, X. Wu
, S. Kim, J. J. Alonso $\$ and H. Pitsch \P

Center for Turbulence Research & Aerospace Computing Lab Stanford University, Stanford, CA

Interactions between different components of a gas turbine are difficult to predict numerically. One of the reasons is the wide range of turbulent scales that need to be modeled. Here, we present an approach to use multiple flow solvers in order to model turbulence appropriately in different sections of a gas turbine. A flow solver based on the Reynoldsaveraged Navier-Stokes (RANS) approach for the turbomachinery sections, while Large-Eddy Simulation (LES) for the combustor is used. These flow solvers run simultaneously and exchange information at the interfaces. In this study we present the approach, some of the validation and report on the application of this approach to a coupled compressorcombustor computation.

I. Introduction

In the development phase of a gas turbine computational fluid dynamics (CFD) is usually used to predict the flow in single components of the engine, such as the compressor, the combustor, or the turbine. The simulation of the *entire* flow path of a gas turbine engine using today's flow solvers is prohibited by the enormous computational costs. However, the increasing computational resources and the improved efficiency of future flow solvers puts the simulation of an entire engine within reach. In order for such a simulation to be useful in the design process it has to deliver accurate results within a reasonable turnover time.

The goal of the Advanced Simulation and Computing Initiative (ASC) of the Department of Energy (DoE) at Stanford is to develop high-performance flow solvers which are able to use highly parallel super-computers for the simulation of an entire engine. While the development of new super-computers is one of the main tasks in the overall ASC effort of the DoE, the physics part of the ASC project at Stanford investigates the development of flow solvers for gas turbine engines in order to improve efficiency, scalability, and modeling of physical effects. However, looking at the wide variety of the flow phenomena, which have to be simulated in the flow path of the engine, it can be seen that only the use of multiple specialized flow solvers, one for the turbo-machinery parts and one for the combustor, can guarantee the efficiency and accuracy of a simulation. The reason for that is, that the flow regimes and the turbulent scales vary dramatically in these two components. Most flow solvers used nowadays in the design process are specialized for one of the two tasks.

The flow field in the turbomachinery portions of the domain is characterized by both high Reynoldsnumbers and high Mach-numbers. The accurate prediction of the flow requires the precise description of the turbulent boundary layers around the rotor and stator blades, including tip gaps and leakage flows. A number of flow solvers that have been developed to deal with this kind of problems have been in use in industry for many years. These flow solvers are typically based on the Reynolds-Averaged Navier-Stokes (RANS) approach. Here, the unsteady flow field is ensemble-averaged removing all dependence on the details

 $^{^* {\}rm Research} \ {\rm Associate}$

[†]Research Associate

[‡]Postdoc

[§]Assistant Professor

[¶]Associate Professor

of the small scale turbulence. A turbulence model becomes necessary to represent the portion of the physical stresses that has been removed during the averaging process. Due to the complexity of the flows in turbomachinery, various parameters in these turbulence models have to be adapted in order to deliver accurate solutions. Since this kind of flow has been the subject of a large number of investigations, these parameters are usually well known and hence, the flow solvers deliver reasonably good results.

The flow in the combustor, on the other hand, is characterized by detached flows, chemical reactions and heat release. The prediction of detached flows and free turbulence is greatly improved using flow solvers based on Large-Eddy Simulations (LES). While the use of LES increases the computational cost, LES has been the only predictive tool able to simulate consistently these complex flows. LES resolves the large scale turbulent motions in time and space and only the influence of the smallest scales, which are usually more



Figure 1. Decomposition of the engine for flow simulations. Compressor¹ and turbine² with RANS; Combustor with LES.³

universal and hence, easier to represent, has to be modeled.^{4,5} Since the energy containing part of the turbulent scales is resolved, a more accurate description of scalar mixing is achieved, leading to improved predictions of the combustion process.⁶ LES flow solvers have been shown in the past to be able to model simple flames and are currently adapted for use in gas turbine combustors.^{7,8}

Here, we want to predict multi-component effects, such as compressor-combustor instabilities, combustorturbine hot-streak migration and combustion instabilities. The flow solvers that describe different components in the gas turbine have to run simultaneously, each computing its part of the domain, and periodically exchanging flow information at the interface (Fig. 1). The simultaneous execution of multiple parallel flow solvers requires the definition of an interface which allows the exchange of flow information and a framework for well-posed boundary conditions in order to process the exchanged data.

The approach to couple multiple simulation codes has already been applied in other areas of application, most notably in global climate simulations,⁹ and found recently more attention in other areas of mechanical engineering.¹⁰ However, the idea to couple RANS and LES flow solvers is a very recent approach and a unique method to construct an LES-RANS hybrid. While other LES-RANS hybrid approaches, such as Detached-Eddy Simulations (DES)¹¹ and Limited-Numerical Scales (LNS)¹² combine LES and RANS in a single flow solver, the approach to couple two existing flow solvers has the distinct advantage to build upon the experience and validation that has been put into the individual codes during their development, and also to run simulations in different domains at different time-steps.

In the current study we want to present the coupling approach and apply it to a compressor-prediffuser geometry of a real aircraft gas turbine engine from Pratt & Whitney. The interface between compressor and the combustor constitutes the upstream interface of a full engine simulation (Fig. 1). The flow leaving the compressor enters first into the prediffuser of the combustor. The function of the prediffuser is to decelerate the flow with a maximum of pressure gain.¹³ For this reason, prediffusers are operated close to the point of flow separation. The flow conditions in the prediffuser ultimately influence the flow split in the combustor and determine the amount of air entering the combustion chamber through the fuel injector. Although the performance of the diffuser is influenced by the flow field leaving the compressor,^{14, 15} little is known about the exact flow features at this location during the design phase of an engine. The reason for this is that the

two components are usually developed in isolation and combined tests are done only in the final prototype assembly.

Here, we will apply the approach of multiple flow solvers to study the flow interactions between these two components. A RANS flow solver computing the final stage of the compressor is coupled with an LES flow solver computing the combustor. The flow in the turbomachinery parts is compressible and governed by the flow around the blades. Hence, a RANS flow solver is an appropriate tool to assess the flow in this section. On the other hand, the prediction of flow separation is facilitated in the LES approach. And while the flow in the current design is not separated, predictions of design modifications have to be able to assess these flow features accurately.

The present paper is organized in the following way:

- 1. We describe the RANS and LES flow solvers as well as the interface and the boundary conditions.
- 2. Two validation studies, one for the interface and one for separated flows in a diffuser are briefly described.
- 3. The application of this approach to a generic compressor/diffuser geometry is shown
- 4. Finally, we demonstrate this approach on a real engine geometry of a Pratt & Whitney gas turbine.

II. Flow Solvers and Interface

In the following we briefly present the computational framework of this study consisting of the flow solvers and the interface. A more comprehensive description of the interface can be found in 16,17

A. RANS Flow Solver

RANS flow solvers are solving the classical Reynolds-Averaged Navier-Stokes equations for turbulent flows. Here, the flow variables are split into a mean and a fluctuating part $u_i = \bar{u}_i + u'_i$, and the Navier-Stokes equations are ensemble-averaged. This delivers a set of equations for the mean velocities, but leaves an unclosed term $\overline{u'_iu'_j}$, which has to be modeled with a turbulence model. Turbulence models are commonly based on the eddy viscosity approach, where the eddy viscosity can be modeled in varying levels of complexity. The most commonly applied models for RANS flow solvers are two-equation models, such as the $k - \epsilon$ or $k - \omega$ models, where two additional transport equations are solved in order to determine the eddy viscosity. In numerical simulations turbo-machinery applications, these models are accepted as a good compromise for between efficiency and accuracy.

The RANS flow solver used for this investigation is the TFLO code developed at the Aerospace Computing Lab (ACL) at Stanford. The flow solver computes the unsteady Reynolds Averaged Navier-Stokes equations using a cell-centered discretization on arbitrary multi-block meshes.¹⁸ The solution procedure is based on efficient explicit modified Runge-Kutta methods with several convergence acceleration techniques such as multi-grid, residual averaging, and local time-stepping. These techniques, multi-grid in particular, provide excellent numerical convergence and fast solution turnaround. Turbulent viscosity is computed from a $k - \omega$ two-equation turbulence model. The dual-time stepping technique^{19,20,21} is used for time-accurate simulations that account for the relative motion of moving parts as well as other sources of flow unsteadiness.

B. LES Flow Solver

LES flow solvers solve for the filtered Navier-Stokes equations. The filter ensures that the large scale turbulence is resolved in time and space resulting in a decomposition of the variables in a resolved and a subgrid part $u_i = \tilde{u}_i + u''_i$. For practical purposes, usually the mesh filter is applied, which means that the cell size defines the filter at each location. Applying the filter to the Navier-Stokes equation leaves an unclosed term $\widetilde{u'_i u''_j}$, which defines the subgrid turbulence. As opposed to the similar unclosed term $\overline{u'_i u'_j}$ from the RANS flow solver, which includes the turbulent motions of all scales, the LES term describes only the subgrid turbulence. With a sufficiently high mesh resolution, the LES solution is rather robust against the chosen subgrid model. Most models use an eddy viscosity approach to model the subgrid stresses. Here, the eddy viscosity can be determined by algebraic models such as the Standard Smagorinsky model,²² or,

as used in this study, by a dynamic procedure, where the solution of the high frequent resolved flow field is used to determine the subgrid stresses.²³

The LES flow solver used for the current study is the CDP- α code developed at the Center for Turbulence Research (CTR) at Stanford. The filtered momentum equations are solved on a cell-centered unstructured mesh and are second-order accurate. An implicit time-advancement is applied. The subgrid stresses are modeled with a dynamic procedure.

C. Interface

Part of the efforts to integrate these flow solvers is the definition of the interface. The optimization of the communication and the processing of the exchanged data to meaningful boundary conditions are some of the challenges encountered. In previous work interface routines have been established and validated with simple geometries.^{24, 25, 17}

The interface used for establishing a connection between the flow solvers consists of routines following an identical algorithm in all flow solvers. The message passing interface MPI is used to create communicators, which are used to communicate data directly between the individual processors of the different flow solvers. This means that each processor of one flow solver can communicate directly with all of the processors of the other flow solvers. This requires the interface routines to be part of the source code of all flow solvers. A detailed description of the common algorithms can be found in Schlüter *et al.*^{26, 17}

In a handshake routine, each processor determines whether its domain contains points on the interface. The location of these points are sent to all processors of the other peer flow solvers. The processors of the peer flow solvers then determine and communicate back, whether the received points are within their own domain. During the actual flow computation all processors communicate data for a common point directly with each other.

The approach of embedding the interface into the source code of each flow solver has been chosen for its efficiency in the communication process. Alternative solutions would be to use a third code, which organizes the communication between the flow solvers, or to limit the peer-to-peer communication to the root processes of each flow solver. While the latter two solutions are usually easier to implement, they cause more communication processes and slow down the computation.

D. Boundary Conditions

The definition of the boundary conditions requires special attention especially on the LES side due to the different mathematical approaches. Since on the LES side part of the turbulent spectrum is resolved, the challenge is to regenerate and preserve the turbulence at the boundaries. At the LES outflow, a body force method has been developed to impose RANS solutions at the outflow of the LES domain.^{27,28}

At the LES inflow boundary, the challenge is to prescribe transient turbulent velocity profiles from ensemble-averaged RANS data. Simply adding random fluctuations to the RANS profiles miss the temporal and spatial correlations of real turbulence and are dissipated very quickly. Instead, a data-base of turbulent fluctuations is created by an auxiliary LES computation of a periodic turbulent pipe flow. The LES inflow boundary condition can then be described as²⁹:

$$u_{i,\text{LES}}(t) = \underbrace{\overline{u}_{i,\text{RANS}}}_{I}(t) + \underbrace{(u_{i,\text{DB}}(t) - \overline{u}_{i,\text{DB}})}_{II} \cdot \underbrace{\frac{\sqrt{u_{(i)}^{\prime 2}}_{\text{RANS}}(t)}{\sqrt{u_{(i)}^{\prime 2}}_{\text{DB}}}}_{III} \tag{1}$$

with the sub-script RANS denoting the solution obtained from the RANS computation and quantities with sub-script DB are from the database. Here, t is the time, u_i stands for the velocity components, and \overline{u}_i is the ensemble average of the velocity component u_i .

Term II of Eq. (1) is the velocity fluctuation of the database. This turbulent fluctuation is scaled to the desired value by multiplication with term III, which ensures that the correct level of velocity fluctuation is recovered.

On the RANS side, inlet and exit boundary condition are applied using the time-averaged solution from the LES side. More advanced boundary conditions are currently under investigation.³⁰

III. Validation

Here, we will present a compilation of the validation work in order to assess the accuracy of integrated RANS-LES computations. An extensive description of the following validation cases can be found in^{17,31}

Interface Validation Α.

As a validation of the interface and the LES inflow boundary condition, a coupled RANS-LES computation of an axisymmetric expansion has been performed. The test-case corresponds to the experimental configuration of Dellenback et al. (1988). Here, a part of the flow domain upstream of the expansion is computed with a RANS code (Fig. 2).

The inlet velocity profiles in the RANS section are specified according to the experimental data at this loca-The RANS flow tion. solver TFLO computes the flow through the upstream pipe and at its outlet transfers the data to the subsequent LES flow solver. The RANS domain is relatively short (0.5D, withD being the diameter of the pipe upstream of the expansion.)

LES flow solver CDP obtains the inflow velocity profiles from the RANS flow solver and specifies its LES inflow boundary conditions according to Eq. (1).

The results of the integrated computation are then validated against the experimental data and against an LES com-



Figure 2. Interface validation: Integrated RANS-LES of a confined jet.



Figure 3. Results of interface validation. Above: axial velocity profiles. Below: Axial velocity fluctuations. Circles: experiments. Solid lines: LES alone defining inflow from experimental data. Dashed lines: Integrated RANS-LES, RANS defining inflow from experimental data, LES defining inflow from simultaneously running RANS.

putation using an inflow data-base at the inlet, in which the data-base statistics are corresponding to the experimental data at the inlet plane.

The RANS mesh contains 350,000 mesh points and is refined near the wall. The LES mesh contains 1.1 million mesh points with the mesh points concentrated near the spreading region of the jet. The far field of the jet is relatively coarse.

Figure 3 shows the LES velocity profiles obtained from this computation. The integrated TFLO-CDP

computation predicts essentially the same results as the single LES computation and matches the experimental data well. Please note that the far field of the jet is not well resolved, and hence, the turbulent fluctuations in the far field are underestimated by both LES computations.

B. Separated Diffuser

For the design of the prediffuser it is crucial to be able to predict flow separation. For this reason, as a validation study, a slightly separated plane diffuser was computed.³¹ The geometry corresponds to that of an experiment by Buice³² and extensive experimental data is available.

In order to assess the accuracy of the LES flow solver and its sensitivity to mesh resolution two different meshes were used. One consist

Figure 5 shows the axial velocity profiles obtained by these two computations. The results are in good agreement with the experimental data. Most importantly, the onset of separation is well predicted. The highly resolved computation demonstrated only slightly better agreement than the the LES computation using a lower resolution.



Figure 4. LES validation: diffuser geometry.

of 2.5M cells and its resolution is equivalent to that in the diffuser of a combustor in the integrated simulation presented below. The other mesh is a refined mesh and consists of 7M cells.



Figure 5. LES validation: mean axial velocity profiles.

IV. Integrated RANS-LES of the NASA Stage 35/Diffuser

In the previous section we have demonstrated the coupled RANS-LES approach on a simple geometry. We have also validated the LES flow solver for slightly separated flows. In the following section we want to increase the complexity of the test-case and demonstrate the value of coupled RANS-LES computations for gas turbine applications.

The test-case is that of the NASA Stage 35 compressor that we extended behind the stators with a diffuser. This geometry is simple enough to study basic flow features, yet it possesses a complexity sought for demonstration purposes. Since no experimental data is available for this flow configuration, the quality of the results has to depend on the validation of each of its compense. Some validation studies of the individual flow solvers are given in Yao *et al.*¹⁸ and Davis *et al.*^{2,33} for the TFLO code and in Mahesh *et al.*³⁴ and Constantinescu *et al.*⁸ for the CDP code. The interface has been developed and tested in the

preceding section as well as in previous work.^{24,17}

The goal of this computation is to demonstrate the feasibility of integrated RANS-LES computations in a turbomachinery environment.

A. Geometry

The compressor geometry for the computed testcase corresponds to that of a modified NASA experimental rig stage 35. The experimental rig consists of a row of 46 rotors and a row of 36 stators. In order to simplify this geometry, the rotor stage has been rescaled to a 36 blade count, which allows to compute an axisymmetric segment of 10° using periodic boundary conditions at the corresponding azimuthal planes.

For this integrated computation, the rotor tipgap has been closed in order to decrease the overall computational costs. The inclusion of the tip-gap is addressed in the TFLO flow solver and poses no additional problem from the integration point of view. The RANS time step was chosen to resolve one blade passing with 50 intervals.

The RANS mesh is a structured multi-block mesh consisting of approximately 1.5 million control volumes. The speed of the rotor was set to a relatively low 5000 RPM in order to keep the flow at the interface within the low-Mach number regime that the LES solver is able to handle. This decrease in rotational speed had to be done for the current case. As shown later, in a real engine, the compresRANS Domain LES Domain 0° 0°

Figure 6. Geometry of coupled NASA stage 35/prediffuser domain.

sor consists of multiple stages resulting in a higher pressure and a higher temperature at the compressor exit. The high temperature of the air in this section of the flow path will ensure that the low-Mach number approximation is not violated, even when the engine is at full load.

The diffuser expands one stator chord length behind the stator. The LES domain starts 1/3 chord behind the stator. The RANS domain reaches 2/3 of the chord length into the LES domain, which essentially means that the RANS outlet plane is just at the expansion of the diffuser.

The diffuser geometry has been chosen with a relatively wide opening such that separation may occur. The diffuser opens towards the centerline of the compressor. Over 3 chord lengths, the diffuser opens up 0.5 chord lengths. The outer wall of the diffuser is straight.

The LES mesh for the CDP flow solver consists of 500,000 control volumes and is concentrated near the walls. LES inflow boundary conditions were defined corresponding to Eq. (1).

In order to initialize the solutions in both domains, separate computations were performed. On the basis of the initial, separate computations, the computational needs for each domain and solver were assessed in order to balance the split of processors for the computation. The load balancing



Figure 7. Integrated RANS-LES of compressor/prediffuser: Velocity distribution at the 50% plane. Close-up of the interface.

between the two flow solvers has to be done manually, since the current version of MPI does not support a dynamic splitting of the processors using multiple codes.

B. Results

The computations were carried out using 64 processors for TFLO and 64 processors for CDP. Eight blade passings were computed in 60 hours of wall clock time using an IBM Power3.

The actual Mach number at the interface was Ma = 0.1 ensuring the validity of the low-Mach number approximation in the LES domain. The mass flux over the interface was conserved with an error of $\approx 0.5\%$.

Figure 7 shows the axial velocity distributions at 50% span of the compressor blades for an instantaneous snapshot of the computation. The upstream RANS solution corresponds to an ensemble averaged solution, while the downstream LES solution is truly unsteady.

The wakes of the stators can clearly be identified in the RANS domain downstream of the stators. The communication of the flow solvers at the interface ensures that the full 3D flow features are transferred from the upstream flow solver to the downstream domain. The boundary conditions of the LES flow solver are defined according to these



Figure 8. Integrated RANS-LES of compressor/prediffuser: Vorticity magnitude distribution at the 50% plane.

data. Hence, the wake of the stator correctly propagates across the interface and can still be found far downstream in the diffuser. It can also be seen that the turbulence, which is resolved in the LES domain, creates a more disturbed velocity distribution.

The differences in the description of turbulence are more apparent in Fig. 8, which shows the vorticity distribution at 50% span of the stator. Here the magnitude of the vorticity is depicted computed according to the unsteady flow field of both domains. In the RANS domain, the vorticity is mainly created due to the mean flow features, such as wall boundary layers, and secondary flows and vortices. The stator creates two vorticity sheets, one on the extrado, one on the intrado. Both vorticity sheets propagate downstream across the interface.

The vorticity distribution in the LES domain is characterized by small scale turbulence. Turbulence present in the upstream RANS domain and modeled by a RANS turbulence model has to be regenerated. The small scale turbulence has been reconstructed at the interface using the LES inflow boundary condition (Eq. (1)). It can be seen that the small-scale turbulence interferes with the stator wakes. The turbulent diffusion of the stator wakes in the RANS domain is modeled with an eddy viscosity model, which gives them a very smooth appearance. In the LES domain, the turbulent transport is given by the resolved turbulence, and hence, vortical turbulent



Figure 9. Velocity profiles in the diffuser; Solid lines: LES only; Dashed lines: RANS-LES; Above: axial velocity; Below: axial velocity fluctuations

structures can be identified. It is obvious that the vortical structure of the turbulence behind the stators may have an influence on the performance of the diffuser. In order to quantify the importance of the integrated RANS-LES computations in diffuser computations of gas turbines, Fig. 9 compares the flow development in the diffuser for two different simulations. The first one (solid lines) is an LES only computation. The inlet velocity profile and the level of turbulence has been specified according to the time-averaged RANS solution at the outlet of the compressor. This solution has been retrieved from the integrated solution and is used to specify the inlet boundary conditions of an LES only computation. The turbulence in this inlet plane is added to the mean velocity profile according to Eq. (1) using the identical turbulence inflow data base as in the integrated RANS-LES. The second set of data (dashed lines) is retrieved from the LES domain of the coupled RANS-LES computation, which means, that at each RANS time step the LES inflow is updated according to the unsteady solution in the compressor. Comparing the velocity profiles, we can see that both solutions are identical in the inlet plane. However, further downstream both solutions are distinctively different.

The profiles of the velocity fluctuations show a similar behavior. At the inlet, both profiles are identical. Here, already shortly downstream the velocity fluctuations are much larger in the integrated RANS-LES computation. This can be explained with the fact that in the integrated RANS-LES unsteady flow features from the compressor are transfered to the LES and deliver unsteady gradients. The production of turbulence is determined as: $P = \overline{u'_i u'_j} \frac{\partial u'_i}{\partial x_j}$. Hence, in the presence of unsteady gradients the turbulence production increases in the LES domain of the integrated RANS-LES. In the current case, the additional turbulence production delivers a different turbulence field which results in a different mean flow field than in the LES only computation. We can conclude that in the current case of the NASA Stage 35/diffuser the use of an integrated RANS-LES can improve the prediction of the diffuser flow.



Figure 11. Isocontours of the axial velocity at the 50% plane near the interface.

V. Pratt & Whitney Engine Geometry

In the previous sections we presented the coupled RANS-LES approach and its application to a simplified compressor geometry. In the final section we want to demonstrate this approach on a real engine geometry.

The geometry considered is that from a Pratt & Whitney aircraft engine (Fig. 10). Here, we present a simulation at the last stage of the high pressure compressor consisting of one rotor and the exit guide vanes (EGV) using the RANS approach. This RANS simulation is coupled with a LES of the prediffuser and the entire combustor. We chose to simulate the entire combustor including the fuel injector, since the flow blockage by the fuel injector and the resulting flow split is considered to be important for the performance of the diffuser.^{14, 13} However, the flow in the combustion chamber is non-reactive, which means that no combustion takes place. The computation of reactive flows has been demonstrated already for this geometry,³ but we consider it as not necessary for the purpose of the present demonstration.

The geometry is a 20° segment of the full engine geometry, which means that we compute one fuel

injector. The blade count of the last stage of the compressor was rescaled to fit the 20° segment, and four rotor blades and seven exit guide vanes are computed in total. The RANS mesh consists of 500,000 cells in a structured multi-block mesh. The combustor mesh consists of 3,000,000 unstructured mesh cells and is refined in the diffuser part.

The computation of 10 blade passings was performed using 128 processors on an IBM SP3. One blade passing needed 10 hours wall clock time. The entire computation was performed within one week.

Fig. 11 shows the axial velocity contours at the interface. As in the previous test-case, the most striking flow features are the wakes of the EGVs entering the diffuser. The flow field is highly turbulent and we can identify coherence of the turbulence due to the wake formation.

VI. Conclusions

In this study we presented an approach to couple two separate flow solvers, one based on the RANS approach, the other based on LES, to improve flow predictions of complex flows.

As an example, we investigated the flow leaving the compressor and entering the diffuser. We have validated the interface and we have validated the LES flow solver for diffuser flows in order to assess the accuracy of such a combined approach.

A computation of a simplified compressor/diffuser geometry demonstrated the value of coupled RANS-LES for this application. Furthermore, this approach was applied to a real engine geometry. The integrated RANS-LES environment provides a computational test bench for the assessment of complex flow interactions, such as that of a compressor/combustor coupling in an aircraft gas turbine engine.

VII. Acknowledgments

We thank the US Department of Energy for the support under the ASC program. We also thank Pratt & Whitney for providing the engine geometry, helpful comments and discussions.

References

¹Schlüter, J. U., Wu, X., Kim, S., Alonso, J. J., and Pitsch, H., "Integrated RANS-LES Computations of Gas Turbines: Compressor-Diffuser," *AIAA Paper*, , No. AIAA 2004-0369, January 2004.

²Davis, R., Yao, J., Clark, J. P., Stetson, G., Alonso, J. J., Jameson, A., Haldeman, C., and Dunn, M., "Unsteady interaction between a transsonic turbine stage and downstream components," *ASME Turbo Expo 2002*, , No. GT-2002-30364, 2002.

 3 Moin, P. and Apte, S., "Large-eddy simulation of realistic gas turbine combustors," AIAA Paper, , No. AIAA 2004-0330, January 2004.

⁴Ferziger, J. H., New Tools in Turbulence Modelling, Vol. New Tools in Turbulence Modelling of Les edition physique, chapter 2 Large eddy simulation: an introduction and perspective, Springer, 1996, pp. 29–47.

⁵Sagaut, P., Large eddy simulation for incompressible flows, Springer, Berlin, 2nd ed., 2002.

⁶Veynante, D. and Poinsot, T., *New Tools in Turbulence Modelling*, chapter 5 Reynolds averaged and large eddy simulation modeling for turbulent combustion, Les edition physique, Springer, 1996, pp. 105–140.

⁷Poinsot, T., Schlüter, J., Lartigue, G., Selle, L., Krebs, W., and Hoffmann, S., "Using large eddy simulations to understand combustion instabilities in gas turbines," *IUTAM Symposium on Turbulent Mixing and Combustion*, 2001, pp. 1–8, Kingston, Canada 2001.

⁸Constantinescu, G., Mahesh, K., Apte, S., Iaccarino, G., Ham, F., and Moin, P., "A new paradigm for simulation of turbulent combustion in realistic gas turbine combustors using LES," *ASME Turbo Expo 2003*, , No. GT2003-38356, 2003.

⁹Trenberth, K. E., *Climate System Modeling*, Cambridge, New York, 1992, chapter 9.

¹⁰Adamidis, P., Beck, A., Becker-Lemgau, U., Ding, Y., Franzke, M., Holthoff, H., Laux, M., Müller, A., Münch, M., Reuter, A., Steckel, B., and Tilch, R., "Steel strip production - a pilot application for coupled simulation with several calculation systems," *Journal of Materials Processing Technology*, , No. 80-81, 1998, pp. 330–336.

¹¹Spalart, P. R., "Trends in Turbulence Treatments," *AIAA paper*, , No. AIAA 2000-2306, 2000, Fluids 2000, June, Denver, CO.

¹²Batten, P., Goldberg, U., and Chakravarthy, S., "LNS-An approach towards embedded LES," *AIAA paper*, , No. AIAA 2002-0427, 2002, 40th AIAA Aerospace Sciences Meeting and Exhibit, January 2002, Reno, NV.

¹³Klein, A., "Characteristics of combustor diffusers," Prog. Aerospace Sci., Vol. 31, 1995, pp. 171–271.

¹⁴Barker, A. G. and Carrotte, J. F., "Influence of compressor exit conditions on combustor annular diffusers, Part 1: Diffuser performance," *Journal of Propulsion and Power*, Vol. 17, No. 3, 2001, pp. 678–686.

¹⁵Barker, A. G. and Carrotte, J. F., "Influence of compressor exit conditions on combustor annular diffusers, Part 2: Flow redistribution," *Journal of Propulsion and Power*, Vol. 17, No. 3, 2001, pp. 687–694.

 16 Schlüter, J. U., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., and Moin, P., "Towards multi-component analysis of gas turbines by CFD: integration of RANS and LES flow solvers," *ASME paper*, , No. ASME GT2003-38350, 2003, ASME Turbo Expo 2003, June 16-19, 2003, Atlanta, GA.

¹⁷Schlüter, J. U., Wu, X., Kim, S., Alonso, J. J., and Pitsch, H., "A Framework for Coupling Reynolds-Averaged with Large Eddy Simulations for Gas Turbine Applications," *Journal of Fluids Engineering*, Vol. submitted, 2004.

¹⁸Yao, J., Jameson, A., Alonso, J. J., and Liu, F., "Development and validation of a massively parallel flow solver for turbomachinery flows," *AIAA paper*, , No. AIAA-00-0882, 2000.

¹⁹Jameson, A., "Time Dependent Calculations Using Multigrid, with Applications to Unsteady Flows Past Airfoils and Wings," AIAA paper, , No. AIAA Paper 91-1596, 1991, AIAA 10th Computational Fluid Dynamics Conference, Honolulu, HI, June 1991.

²⁰Alonso, J. J., Martinelli, L., and Jameson, A., "Multigrid Unsteady Navier-Stokes Calculations with Aeroelastic Applications," AIAA Paper, , No. AIAA 95-0048, 1995, AIAA 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, 1995.

²¹Belov, A., Martinelli, L., and Jameson, A., "Three-Dimensional Computations of Time-Dependent Incompressible Flows with an Implicit Multigrid-Driven Algorithm on Parallel Computers," *Proceedings of the 15th International Conference on Numerical Methods in Fluid Dynamics, Monterey, CA*, 1996.

²²Smagorinsky, J., "General circulation experiments with the primitive equations, I, the basic experiment," *Mon. Weather Rev.*, Vol. 91, No. 3, 1963, pp. 99–152.

²³Germano, M., Piomelli, U., Moin, P., and Cabot, W., "A dynamic subgrid-scale eddy viscosity model," *Phys. Fluids*, Vol. A, No. 3 (7), July 1991, pp. 1760–1765.

²⁴Shankaran, S., Liou, M.-F., Liu, N.-S., and anf J. J. Alonso, R. D., "A multi-code-coupling interface for combustor/turbomachinery simulations," *AIAA paper*, , No. AIAA 2001-0974, 2001.

²⁵Schlüter, J. U., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., and Moin, P., "Integration of RANS and LES flow solvers for simultaneous flow computations," *AIAA paper*, , No. AIAA 2003-0085, 2003.

²⁶Schlüter, J. U., Pitsch, H., and Moin, P., "Boundary Conditions for LES in Coupled Simulations," AIAA paper, , No. AIAA-2003-0069, 2003.

²⁷Schlüter, J. U., Pitsch, H., and Moin, P., "Consistent Boundary Conditions for Integrated LES/RANS Simulations: LES Outflow Conditions," *AIAA paper*, , No. 2002-3121, 2002, 32nd AIAA Fluid Dynamics Conference, June 24-27, St, Louis, MO.

²⁸Schlüter, J. U., Pitsch, H., and Moin, P., "LES Outflow Conditions for Integrated LES/RANS Simulations," accepted for publication to AIAA Journal, 2004.

²⁹Schlüter, J. U., Pitsch, H., and Moin, P., "Large Eddy Simulation inflow conditions for coupling with Reynolds-averaged Flow Solvers," *AIAA Journal*, Vol. 42, No. 3, March 2004, pp. 478–484.

³⁰Kim, S., Schlüter, J., Wu, X., Alonso, J. J., and Pitsch, H., "Integrated simulations for multi-component analysis of gas turbines: RANS boundary conditions," *AIAA Paper*, , No. AIAA 2004-3415, July 2004.

³¹Schlüter, J., Wu, X., and Pitsch, H., "LES of a separated plane diffuser," *AIAA Paper*, 2005, accepted at 43rd Aerospace Sciences Meeting, Reno 2005.

³²Buice, C. U., *Experimental investigation of flow through an asymmetric plane diffuser*, Ph.D. thesis, Stanford University, August 1997.

³³Davis, R., Yao, J., Alonso, J. J., Paolillo, R., and Sharma, O. P., "Prediction of main/secondary-air system flow interaction in a high pressure turbine." *AIAA Paper*, , No. AIAA-2003-4833, 2003.

³⁴Mahesh, K., Constantinescu, G., Apte, S., Iaccarino, G., and Moin, P., "Large-eddy simulations of gas turbine combustors," *Annual Research Briefs 2001*, 2001, pp. 3–18, Center for Turbulence Research, NASA Ames/Stanford Univ.